



Numerical Calculations of Wind Flow in a Full-Scale Wind Test Facility

C. H. Oh
J. M. Lacy

June 20, 1999 – June 25, 1999

10th International Conference on Wind Engineering

This is a preprint of a paper intended for publication in a journal or proceedings. Since changes may be made before publication, this preprint should not be cited or reproduced without permission of the author.

This document was prepared as a account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, or any of their employees, makes any warranty, expressed or implied, or assumes any legal liability or responsibility for any third party's use, or the results of such use, of any information, apparatus, product or process disclosed in this report, or represents that its use by such third party would not infringe privately owned rights. The views expressed in this paper are not necessarily those of the U.S. Government or the sponsoring agency.



Numerical Calculations of Wind Flow in a Full-Scale Wind Test Facility

C. H. Oh and J. M. Lacy

Idaho National Engineering and Environmental Laboratory

Idaho Falls, ID 83415-3885

USA

ABSTRACT: Numerical studies on wind flow around the Texas Tech University (TTU) Wind Engineering Research Field Laboratory (WERFL) building were conducted. The main focus of this paper is wind loads on the TTU building in the INEEL proposed Windstorm Simulation Center. The results are presented in the form of distributions of static pressure, dynamic pressure, pressure coefficients, and velocity vectors on the surface and the vicinity of the TTU building.

1 INTRODUCTION

The Idaho National Engineering and Environmental Laboratory (INEEL), a U.S. Department of Energy research facility, is proposing a concept for a structural wind testing facility called the Windstorm Simulation Center (WSC) (Figures 1 and 2). This facility would direct wind gusting up to 80 m/s at full-scale two-story houses, light industrial buildings, and similar structures to develop, validate, and demonstrate inexpensive new durability and energy efficiency technologies. Innovations developed and demonstrated at the WSC would be used to economically improve the safety of houses and other pre-engineered structures.

The WSC concept is an open-circuit, open-jet wind tunnel. The 28-fan main drive system is located upstream of the test section to prevent damage from test specimen debris. Planetary boundary layer wind turbulence will be generated actively by control of the main drive and a vertically-oriented random-motion airfoil cascade upstream of the test section.

Prior to requesting funding for the WSC, the INEEL will demonstrate the concept and widespread benefits of a large-scale structural wind test facility with a smaller Pilot Windstorm Center (PWC). The PWC will generate winds gusting to 55 m/s on smaller single-story structures such as

manufactured, modular, and modest site-built housing.

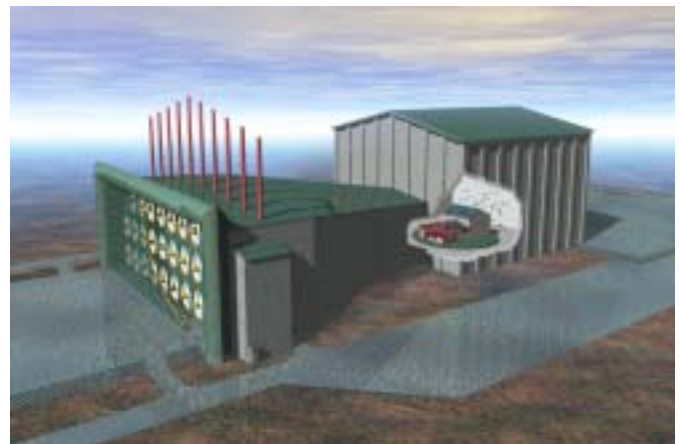


Figure 1. INEEL's proposed Windstorm Simulation Center.

In conjunction with construction of the PWC, a \$23M research program has been proposed to the USDOE to provide funding to both federal and university researchers to develop and validate cost-effective technologies to improve the safety and energy efficiency of affordable housing.

Preliminary design studies have been initiated to study concepts for both the pilot and full-size wind test facilities. This paper reports the results of a numerical study to characterize flow through the full-size WSC and over a test building, and the subsequent wind loads on the test specimen.

Experimental studies are also currently underway, and will be reported at a later date.

The test specimen was chosen to represent Texas Tech University's WERFL building, a heavily

instrumented low-rise structure for which extensive correlated wind velocity and building surface pressure data is available.

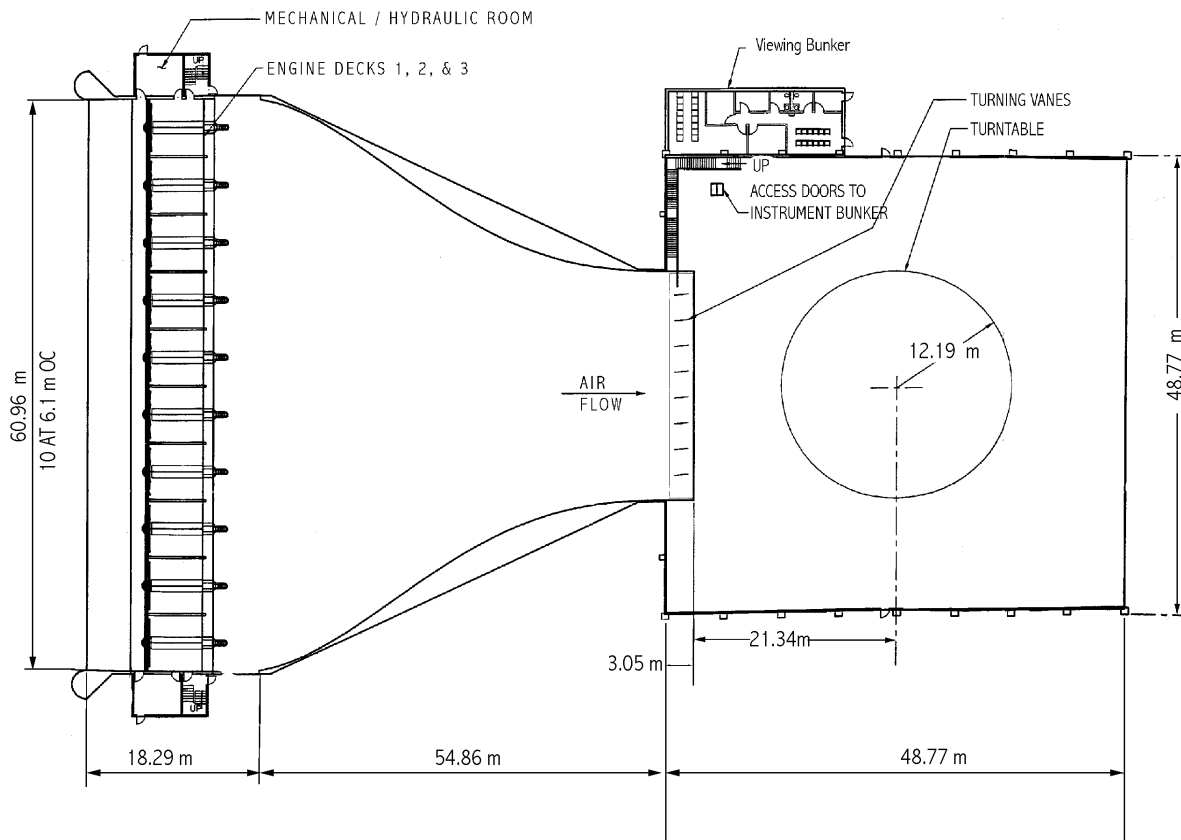


Figure 2. Top view of the WSC.

A finite volume formulation of renormalized group (RNG) turbulence model was applied to calculate three-dimensional, turbulent flows through the WSC facility and around the test specimen. The pressure drops from this calculation account for all pressure losses in the tunnel including friction, expansions, contractions, and velocity changes. The results are presented in the form of pressure distribution, pressure coefficient distribution, velocity distribution and energy ratio. Recommendations are presented for optimizing the facility geometry to minimize main drive power required, while maintaining representative flow and associated pressures around the test specimen.

2 PROBLEM DESCRIPTION

This problem consists of three-dimensional turbulent (Re at the throat is 1.0×10^8) flow through the wind tunnel as depicted in Figure 3. These calculations take advantage of the wind tunnel's symmetry about a vertical plane along its axial centerline. The inlet half-width is 30.48 m; overall length is 119.9 m; the height of the inlet is 18.3 m; and the height of the test section is 26.82 m. Dimensions of the test building are 6.85 m wide (from the vertical centerline), 9.1 m long, and 4 m high.

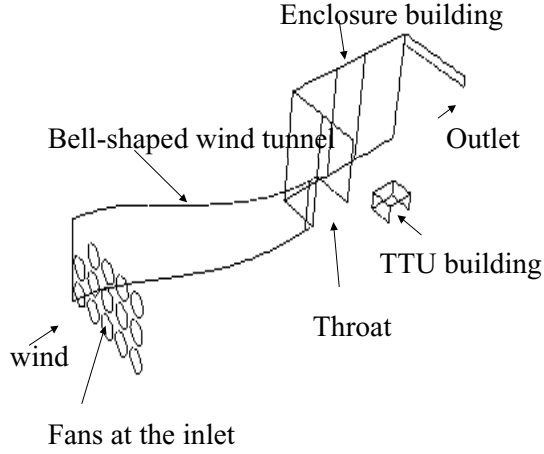


Figure 3. WSC model using symmetry at the vertical centerline.

3 GOVERNING EQUATIONS

The flow field in the wind tunnel is described by the steady state mass, and momentum equations;

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

$$\frac{\partial}{\partial x}(\rho u u_i) = -\frac{\partial P}{\partial x} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho g_i + F_i \quad (2)$$

where u_i is the time-mean velocity, ρ is the fluid density, P is the static pressure, μ is the dynamic viscosity of the fluid, and F_i are external body forces.

The numerical solution of the governing partial differential equations (eqs 1-2) begins with discretization of the field into a collection of control volumes. The differential equations are approximated by a set of algebraic equations on this collection, which is then solved to produce a set of discrete values that approximate the solution of the partial differential system over the field. The governing equations can be rewritten in a canonical form as

$$\frac{\partial}{\partial x_j}(\rho u_j \Phi) = \frac{\partial}{\partial x_j}(\Gamma \frac{\partial \Phi}{\partial x_j}) + S \quad (3)$$

where the generic dependent variable Φ , diffusion coefficient Γ , and source term S can be specified to uniquely define a particular equation. In this study, the algebraic equation system is solved by the algorithm called SIMPLE (Patanka, 1980), a semi-implicit iterative scheme which starts from an initial guess and converges to a solution after performing a sufficient number of iterations. Of the several commercial computational fluid dynamics software packages, the FLUENT (FLUENT, 1998) code was used for these calculations. Flows, geometry definitions, boundary conditions specifications, and post-processing routines were modified to fulfill the modeling needs of this research. Validity of the FLUENT numerical code has been verified by several supporting studies (FLUENT, 1993).

Turbulence affects the flow calculations via the effective viscosity, defined by the relation

$$\mu_{eff} = \mu + \mu_t \quad (4)$$

where μ is the molecular viscosity and μ_t is the turbulent viscosity calculated in this study by using the renormalization group (RNG) $k - \epsilon$ turbulence model (Yakot and Orszag, 1986) the RNG-based $k - \epsilon$ turbulence model follows the two-equation turbulence modeling framework and uses dynamic scaling and invariance together with iterated perturbation methods. The steady state transport equations for k and ϵ in the RNG model are

$$u_i \frac{\partial k}{\partial x_i} = \nu S^2 - \epsilon + \frac{\partial}{\partial x_i} \alpha \nu_i \frac{\partial k}{\partial x_i} \quad (5)$$

$$u_i \frac{\partial \epsilon}{\partial x_i} = c_1 \frac{\epsilon}{k} \nu S^2 - c_2 \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_i} \alpha \nu_i \frac{\partial \epsilon}{\partial x_i} - R \quad (6)$$

The rate of strain term R is given by

$$R = \frac{0.09 \eta^3 (1 - \eta/\eta_0) \epsilon^2}{1 + 0.012 \eta^3} \frac{1}{k} \quad (7)$$

where $\eta = S k / \epsilon$, η_0 , c_1 , and c_2 are 4.38, 1.42, and 1.68, respectively.

4 BOUNDARY CONDITIONS AND MODEL DEVELOPMENT

The model consists of a wind tunnel, an enclosure building at the exit of the wind tunnel, and TTU building inside of the enclosure building which is

located at 19.83 m from the wind tunnel nozzle throat.

The inlet geometry was modeled as 14 multiple fans in a 3 row by 5 column array (the top row has 4 fans). The outlet condition of the enclosure building is atmospheric pressure. The total number of computational cells used in this study is 163,050. Much finer meshes were incorporated near the wind tunnel wall boundaries and along the TTU building surfaces to maintain y^+ values near 30.

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon large flow field. The fan boundary allows us to input an empirical fan curve, which governs the relationship between head (pressure rise), and flow rate (velocity) across a fan element. In this study, pressure jump across the fan was inputted from an iterative calculation until it reaches a desired velocity value at the wind tunnel nozzle throat. At the entire wall boundary and surfaces of the TTU building, no slip boundary condition was applied. Air at 27⁰ C and 86160 Pa is essentially constant density and eliminates the need for buoyancy calculations. The entire wind tunnel, enclosure building, and the TTU building were modeled as a three-dimensional symmetry at the centerline.

Note that no additional turbulence via fan control or airfoil cascade is introduced into the flow for this study.

5 RESULTS AND DISCUSSIONS

Figure 4 shows velocity vectors in the enclosure building. The velocity vectors plotted in the Figures 4, 5, and 6 represent velocities at the center of each control volume. For a better presentation, the velocity vectors in the flow domain were plotted only on the every 5th cells in the vertical and horizontal coordinates. Figure 4 shows that axial velocity is accelerated at the nozzle throat due to the area contraction. The vectors are deflected when they reach the TTU building and flow towards a 45 degree angle to exit out.

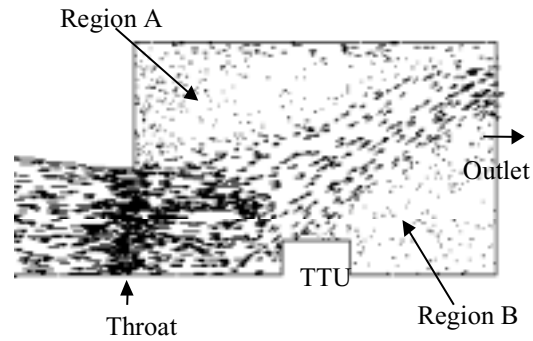


Figure 4. WSC model.

Figures 5 and 6 shows the detailed flow pattern in the upper front section of the enclosure building and the rear portion of the TTU building. The flow recirculation is apparent in the counter-clockwise in region A due to the upstream jet flows. This upstream jet flow also causes air entertainment behind the TTU building as shown in Figure 6. These flow patterns ultimately help us to design the WSC in an efficient manner.

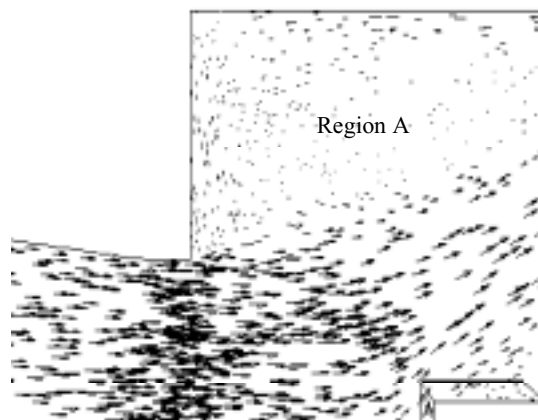


Figure 5. Flow pattern in region A.

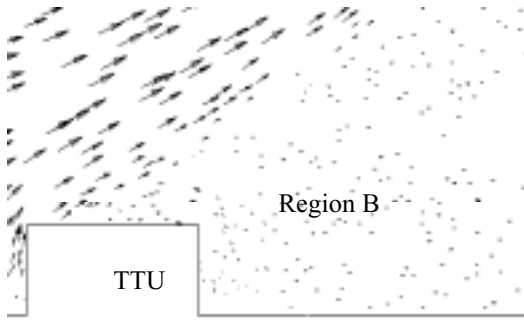


Figure 6. Flow pattern in region B.

Figure 7 shows static pressure distributions on the TTU building. The static pressure is the maximum at the building front where the air flow is stagnated. Also the static pressures are lower on the top and sides than the barometric pressure as the flow accelerates around the corner. The back side of the TTU building has low pressure inducing air entrainment shown in the previous Figure 6.

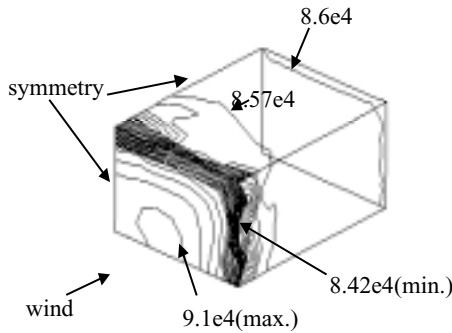


Figure 7. Static pressure on TTU building.

Figure 8 shows dynamic pressure distributions on the TTU building. The dynamic pressure is the minimum at the front due to the flow stagnation. The dynamic pressure is the highest at the corner as the flow accelerates around the corners. As shown figures 7 and 8, the flow characteristics cause the high and low pressure on the building. These pressure results are consistent with our expectations.

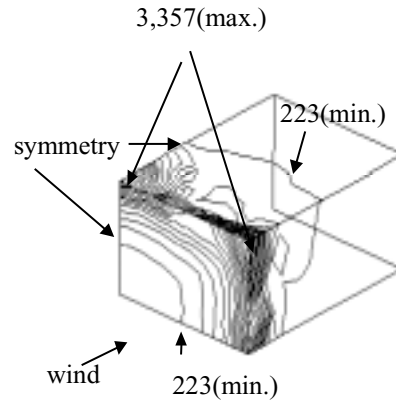


Figure 8. Dynamic pressure on TTU building.

Figure 9 shows pressure coefficient defined as the following relationship:

$$\text{pressure coefficient} = \frac{P - P_o}{\frac{\rho U^2}{2}} \quad (8)$$

where P is the measure surface pressure, P_o is the reference pressure, ρ is the air density, and U is the reference wind speed. The pressure coefficient is low at the corner and side because flow is accelerated around corners where static pressure is low. These pressure coefficients are consistent with those referenced for the square building (R. Blevins, 1984).

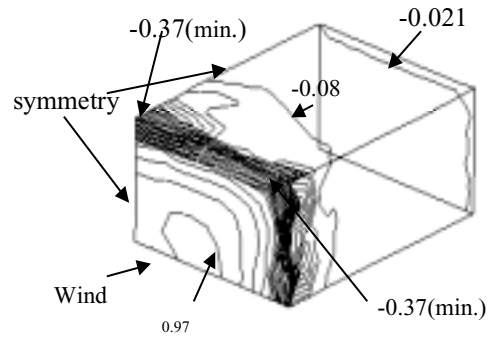


Figure 9. Pressure coefficient on TTU building.

6 CONCLUSION

Pressure results are consistent with our expectations, which will be validated from the future study. Also, the pressure coefficients

obtained from study are consistent with those cited in the reference for the square building. In overall, CFD code can be a useful tool to design the WSC.

7 RECOMMENDATIONS FOR FUTURE STUDIES

1. Further numerical studies to optimize flow and energy consumption are necessary.
2. An attempt should be made to incorporate the effect of turbulence generating devices into this model.
3. Experimental studies have been initiated in a 1/14th scale (105 kW) model (Figure 10). These should include an effort to validate these numerical results with respect to energy consumption and pressure distribution on representative test structures.
4. Large eddy simulation (LES) turbulence model will be used to determine the transient behavior of wind loads on a representative test structure.



Figure 10: 1/14th scale model WSC

REFERENCES

Blevins, R.D. 1984. *Applied Fluid Dynamics Handbook*. New York, NY. Van Nostrand Reinhold Company Inc.

FLUENT 5 User's Guide 1998. Fluent Inc.:Lebanon, NH.

FLUENT V4.2 Validation Problems 1993. Fluent Inc.: Lebanon, NH.

Patanka, S.V. 1980. *Numerical Heat Transfer and Fluid Flow*. Washington, DC: Hemisphere

Yakhot, V. & S.A. Orszag 1986. Renormalization group analysis of turbulence. I basic theory. *J. Sci. Comput.* 1 (1):3-51.

ACKNOWLEDGMENT

This work was performed under the auspices of the U.S. Department of Energy under the DOE Operations Office Contract No. DE-AC0794ID13223.